



# OptiSPICE

Opto-Electronic Circuit Design  
Software

6.0

## New Features

---

OptiSPICE is the first circuit design software for the analysis of integrated circuits including interactions of optical and electronic components. It allows for the design and simulation of opto-electronic circuits at the transistor level, from laser drivers to transimpedance amplifiers, optical interconnects and electronic equalizers.





## Key Features for OptiSPICE 6.0

**OptiSPICE 6.0** includes several enhancements including visualizers, model and device improvements. Key new features include:

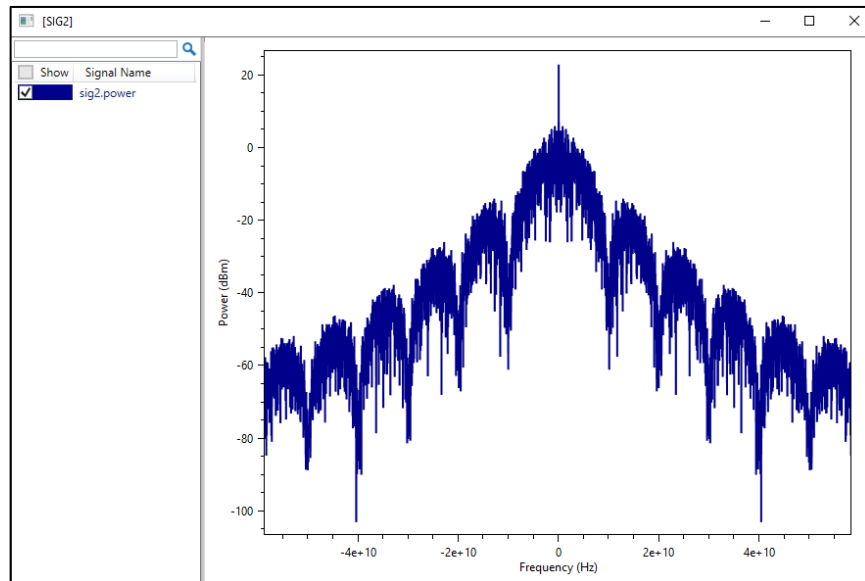
- Three new visualizers have been added to OptiSPICE library, Electrical Spectrum analyzer, Electrical Eye Diagram Analyzer and Optical Spectrum Analyzer.
- Electrical Spectrum Analyzer performs Fourier Transform on time domain simulation results for a given node voltage.
- Optical Spectrum Analyzer performs Fourier Transform on time domain simulation results for a given optical signal.
- All the simulation results from probes placed in the schematic are exported as a single HDF5 files in addition to XML and text. HDF5 file import is supported by languages like Matlab and Python. With this feature users can easily create scripts for processing simulation results.
- Bit generator has been enhanced to allow the import of user defined bit sequence from a text file, generate a random bit sequence based on a user defined seed.
- The filter parameter extractor tool has been improved to provide a better fit.
- A new electrical filter component has been added to the electrical device library. It is designed to accept the PRF file generated by the filter parameter extractor.

## New Visualizers

### Electrical Spectrum Analyzer:

A new electrical spectrum analyzer device has been added to the Visualizers library. This visualizer allows the user to inspect the spectral response of a time domain simulation result for an electrical signal. Its output format, axis units and scales can be adjusted by the user through the properties panel.

An example circuit demonstrating the capabilities of this new visualizer can also be found in *OptiSPICE 6.0 samples\Visualizers\Electrical\Spectrum Analyzer\*



**Fig 1: Electrical Spectrum Analyzer** – The Electrical Spectrum Analyzer is configured to show the power spectrum of an electrical signal in dBm scale. It shows the response for both positive and negative frequencies in Hz.

### Electrical Eye Diagram Analyzer:

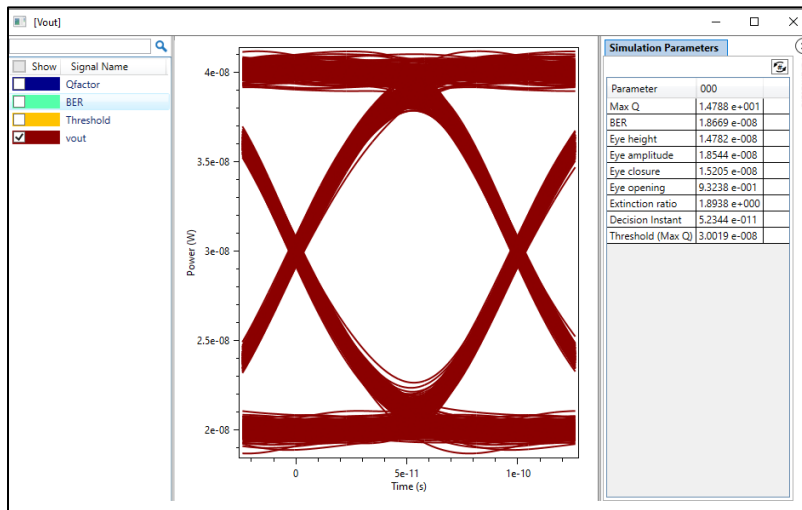
A new electrical eye diagram analyzer device has been added to the Visualizers library. This visualizer takes an electrical signal and a bit stream as an input and generates an eye diagram and related statistics like, bit error rate, Q-factor, eye height, eye closure and extinction ratio.

An example circuit demonstrating the capabilities of this new visualizer can also be found in *OptiSPICE 6.0 samples\Visualizers\Electrical\Eye Diagram\*



**OptiSPICE**

Opto-Electronic Circuit Design  
Software

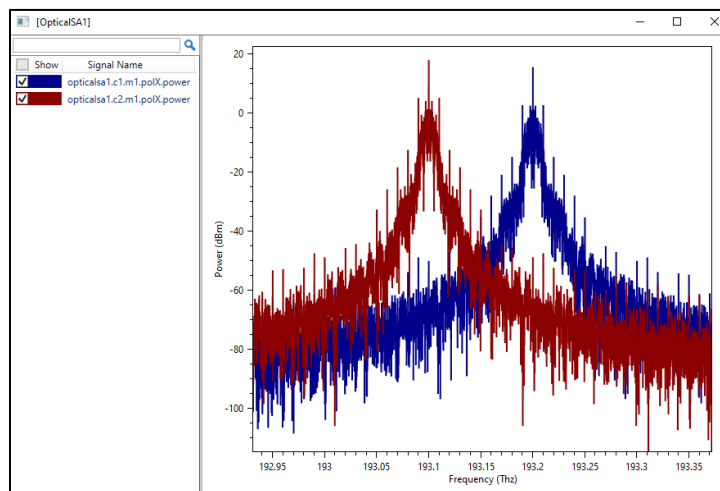


**Fig 2: Eye Diagram Analyzer** – In this example, a transient electrical signal paired with its corresponding bit stream produces the eye diagram and statistics related to signal quality.

### **Optical Spectrum Analyzer:**

A new optical spectrum analyzer device has been added to the Visualizers library. This visualizer allows the user to inspect the spectral response of a time domain simulation result for an optical signal. Its output format, axis units and scales can be adjusted by the user through the properties panel. It can show the results for multiple channels in the same plot.

An example circuit demonstrating the capabilities of this new visualizer can also be found in *OptiSPICE 6.0 samples\Visualizers\Optical\Spectrum Analyzer\*



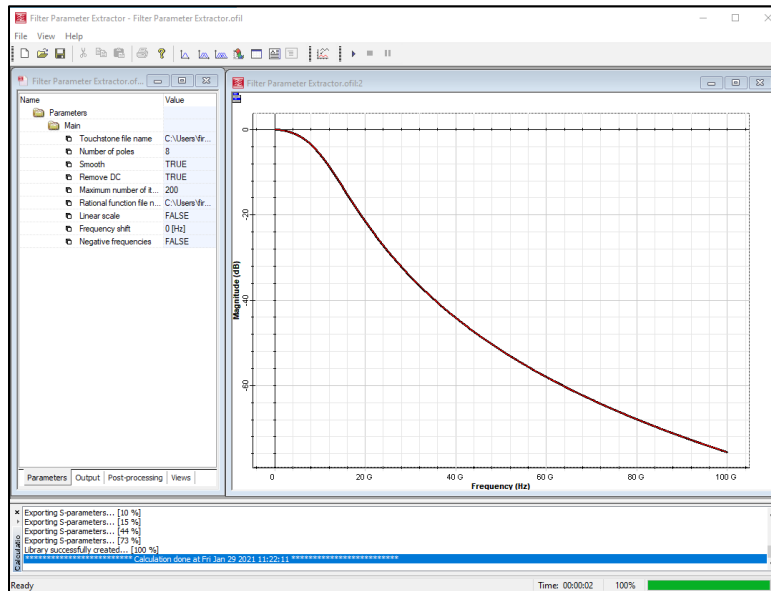
**Fig 3: Optical Spectrum Analyzer** – In this example the Electrical Spectrum Analyzer is configured to show the power spectrum of an electrical signal in dBm scale. It shows the response for both positive and negative frequencies in Hz.



## Updated Electrical Filter Modeling

### Improved Fitting Algorithm:

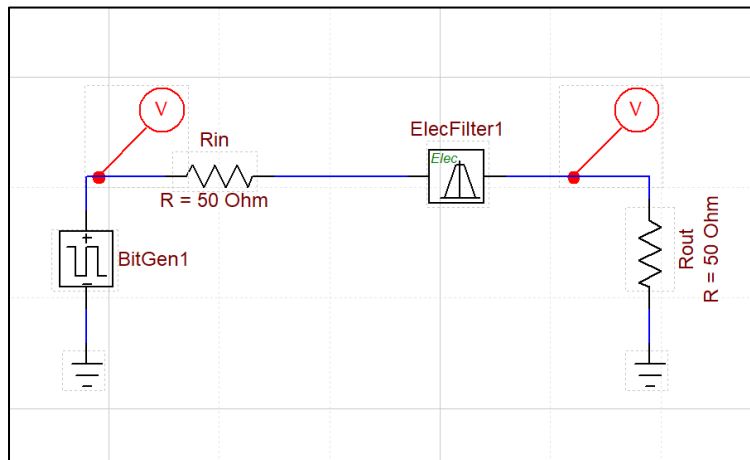
The fitting algorithm for the filter parameter extractor has been modified to vary the number of poles until a good fit is achieved. This improves the fitting accuracy in frequency domain and results in an improved time domain representation for the filter.



**Fig 4: Filter Parameter Extractor** – The Electrical Spectrum Analyzer is configured to show the power spectrum of an electrical signal in dBm scale. It shows the response for both positive and negative frequencies in Hz.

### New Filter Element:

A new electrical filter element has been added to the electrical devices library. It takes the filter model created by the filter parameter extractor as an input and generates a time domain model representation of the filter.

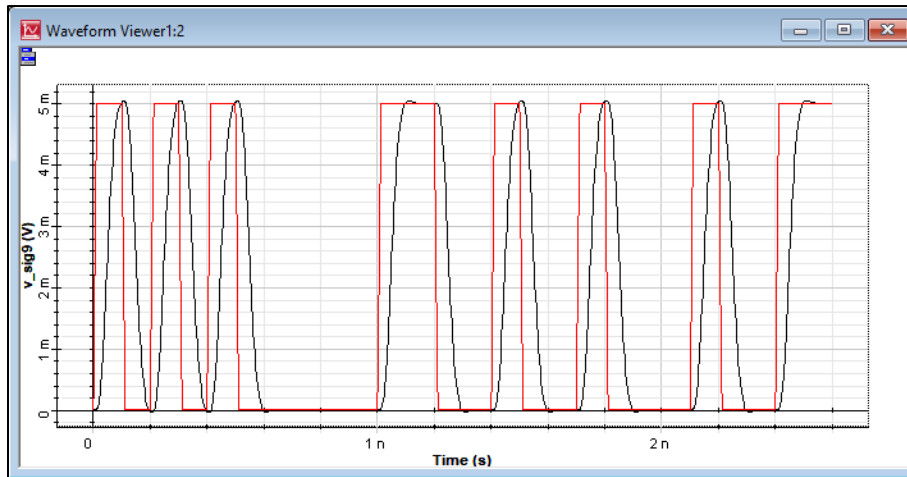


**Fig 5: Electrical Filter** – In this example, signal created by the bit generator is passed through a low pass electrical filter model generated by the filter parameter extractor.



**OptiSPICE**

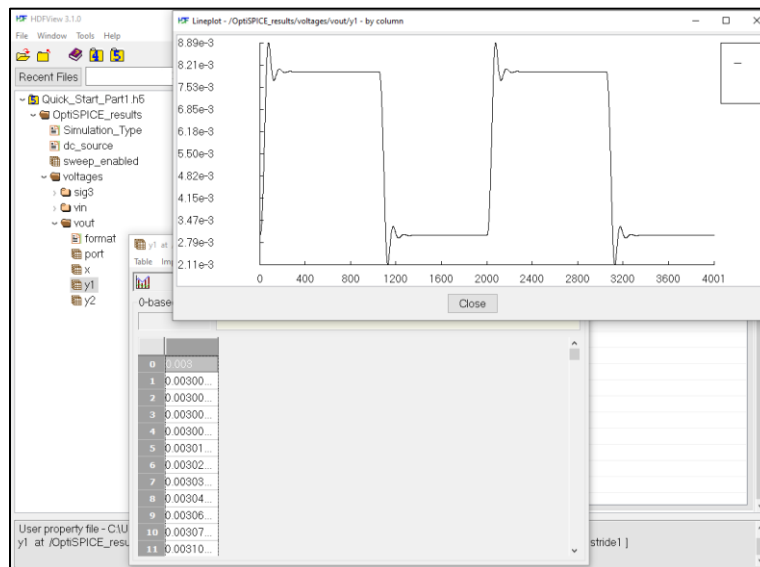
Opto-Electronic Circuit Design  
Software



**Fig 6: Time Domain Filter Modeling** – The signals generated by the bit stream (red) is passed through a low pass filter. The simulation result of the low pass filter (black) is shown in this figure.

### Simulation Results in HDF5 Format

Support for HDF5 data format was added to OptiSPICE simulation engine. Now, the simulation results are exported as a single HDF5 file in addition to ASCII format and XML header file. HDF5 data format is supported by many programming and scripting languages like C/C++, Python and Matlab. HDF5 data format provides additional flexibility to the user in selecting the right environment for processing the results generated by the simulator.



**Fig 7: HDF5 Data Export** – The image above shows the simulation results exported by OptiSPICE in HDFView.

